Numerical simulation of the fountain flow instability in injection molding

M.G.H.M. Baltussen, M.A. Hulsen*, G.W.M. Peters

Department of Mechanical Engineering, Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, The Netherlands

A R T I C L E   I N F O

Article history:
Received 16 December 2008
Received in revised form 4 March 2010
Accepted 9 March 2010

PACS:
47.50.Gv
47.55.dr
47.50.Gj
47.20.Gv

Keywords:
Injection molding
Fountain flow
Viscoelastic instability
Finite element method

A B S T R A C T

For the first time, the viscoelastic flow front instability is studied in the full non-linear regime by numerical simulation. A two-component viscoelastic numerical model is developed which can predict fountain flow behavior in a two-dimensional cavity. The eXtended Pom-Pom (XPP) viscoelastic model is used. The levelset method is used for modeling the two-component flow of polymer and gas. The difficulties arising from the three-phase contact point modeling are addressed, and solved by treating the wall as an interface and the gas as a compressible fluid with a low viscosity. The resulting set of equations is solved in a decoupled way using a finite element formulation. Since the model for the polymer does not contain a solvent viscosity, the time discretized evolution equation for the conformation tensor is substituted into the momentum balance in order to obtain a Stokes like equation for computing the velocity and pressure at the new time level. Weissenberg numbers range from 0.1 to 10. The simulations reveal a symmetric fountain flow for Wi = 0.1–5. For Wi = 10 however, an oscillating motion of the fountain flow is found with a spatial period of three times the channel height, which corresponds to experimental observations.

© 2010 Elsevier B.V. All rights reserved.

1. Introduction

One of the most prominent flow effect which occurs in injection molding is fountain flow [1], already observed by West in 1911 for a mercury filled capillary [2]. This effect occurs during filling near the polymer melt flow front. Much attention was paid to fountain flow kinetics [3,4], since it plays an important role in melt solidification and molecular orientation [5,6]. Both influence the final product properties.

The movement of the contact point is a difficult problem. In a Lagrangian or ALE framework the free surface is a moving boundary of the domain and the contact is relatively easy handled by assuming, for example, sticky conditions (velocity of the fluid becomes equal to the velocity of the wall) if a node on the surface tries to cross the wall. This is nicely described in a recent paper by Dimakopoulos and Tsamopoulos [7] (see also [8–10]). In a Lagrangian or ALE framework the interface between the two phases (liquid and solid) is aligned with element boundaries. A drawback of these approaches is that elements can become too distorted and remeshing is required if there is a large deformation of the interface. Since in injection molding very large deformation is achieved we investigate in this paper a different approach using an implicit interface description in an Eulerian framework. For the interface description we use a levelset function. The approach is similar to [11], where a discontinuous color function has been used instead of a levelset function. Since the interface position is implicit, the free surface is not a boundary of the domain and we need to consider both the fluid domain and the outside domain. The latter is modeled as a fluid with a low viscosity. An immediate drawback of this approach is that the contact point is now a three-phase contact point in an Eulerian framework with all its numerical difficulties, such as possible gas inclusion due to inaccurate description of the interface. However a large deformation limit due to excessive element distortions does not exist.

During injection molding several defects can occur, one of them being alternating rough and dull bands on the surface of a finished product. The reason for this type of defect to occur is the time periodic asymmetric motion of the fountain flow [12–14], which has been shown to be a viscoelastic instability [15–17]. Bogaers et al. [17] studied the linear stability of a viscoelastic fluid at relatively low Weissenberg numbers and hence only predicted the critical conditions for which the flow becomes unstable. The non-linear motion of the flow cannot be described by this analysis. Deformation rates (|γ|) in injection molding are very high, 10^4 s⁻¹, and the typical time scale related to the viscoelastic material is Ψ1/(2η), which is of the order of 10⁻² s. This leads to Weissenberg numbers of O(100). With the development of the log-conformation method by Fattal and Kupferman [18] and the implementation using a finite element method by Hulsen et al. [19] and Yoon and Kwon [20], the
range of attainable Weissenberg numbers has grown by one or two decades, in some benchmark problems. In addition to the development of numerical methods, advanced constitutive models have been developed for polymer melts, amongst which the eXtended Pom-Pom model (XPP) [21]. This model gives a good description of the rheology in shear as well as in elongation [21,22].

The aforementioned progress in numerical techniques and constitutive models has opened the road to numerical simulations at flow conditions which are closer to processing conditions. The objective in this paper is the simulation of non-linear viscoelastic effects during the filling of a simplified injection molding geometry, including viscoelastic instabilities. The paper is structured as follows. First, the problem domain and governing equations will be given in Section 2. Next, the numerical implementation of this model will be discussed in Section 3 and the simulation results will be presented in Section 4. Finally, conclusions will be given in Section 5.

2. Model

2.1. Geometry

The filling of a two-dimensional channel is modeled. Initially the channel is partly filled with a rectangular block of polymer, which is moved by a piston into the empty part of the channel, see Fig. 1a. In order to avoid problems with modeling a moving flow domain, the piston velocity is subtracted from the velocity field, see Fig. 1b.

Note that two singularities are introduced in the flow field at the corners of the piston due to the jump in velocity between the piston and the wall. Two subdomains are present: the melt domain \( \Omega_m \) and the gas domain \( \Omega_g \), see Fig. 2, with the piston boundary \( \Gamma_p \), the moving wall of the polymer domain \( \Gamma_{wp} \), the moving wall of the gas domain \( \Gamma_{wg} \), an inflow boundary of the gas \( \Gamma_{gin} \) and the flow front \( \Gamma_f \). The position of the flow front is part of the solution and not known a priori.

2.2. Modeling of the gas domain

The approach adopted is to model the gas domain as a (very) low viscosity viscous fluid. As was pointed out by Dussan [23], the flow of two incompressible fluids with a different viscosity and stick boundary conditions leads to a streamline pattern shown in Fig. 3. At the flow front two vortices are observed in the low-viscosity (gas) domain. Gas traveling from the right to the contact point (c) cannot leave to the left and therefore has to leave to the right causing the vortices.

The vortices result in large velocity gradients near the interface, which can lead to large stresses. If these large stresses are of the same order of magnitude as the stresses in the melt, the shape of the flow front changes due to the stresses in the gas. Therefore the polymer/gas viscosity ratio should be taken sufficiently large. Different solutions to the undesired motion of the flow front are possible and all aim to lower the undesirable stresses. Recall that the reason for the vortices are the stick boundary conditions in combination with the incompressibility condition. By allowing slip at the gas part of the wall [11], the vortices disappear, see Fig. 4.

This is probably another reason for the success of the slip boundary condition in mold filling simulations. The downside of this method is that the exact position of the contact point has to be known, and that the type of the boundary condition is position dependent. Both of them are non-trivial problems. Another option is to model the gas as a compressible fluid, also leading to removal of the vortices, see Fig. 4 (b). It should be noted that the gas domain is fictitious and it is merely present to allow the flow front to develop and move as a traction-free surface, while not causing undesired flow front motion by developing too high stresses. In this work the gas domain will be modeled as a compressible (fictitious) viscous fluid.

2.3. Governing equations

The flow instabilities which are studied are known to be of a viscoelastic nature and are not thermally induced [17]. Therefore the problem is modeled as an isothermal flow. Due to the high viscosity of polymer melts, the viscous forces dominate the problem and inertial and gravitational effects are negligible as well. The melt is modeled as an incompressible viscoelastic fluid. The gas is modeled as a compressible viscous fluid, as discussed in the previous subsection. This results in the following momentum and continuity equations:

\[
- \nabla \cdot \sigma = 0 \quad \text{in} \ \Omega \\
\nabla \cdot \mathbf{u} = 0 \quad \text{in} \ \Omega_m
\]
where $\sigma$ is the Cauchy stress tensor, $\mathbf{u}$ is the velocity vector and $\Omega = \Omega_m \cup \Omega_g$. The constitutive equations for the Cauchy stress tensor for the polymer and gas domain are:

$$\sigma = -p\mathbf{I} + \mathbf{\tau} \quad \text{in} \quad \Omega_m$$

$$\sigma = -p\mathbf{I} + 2\eta_g\mathbf{D} \quad \text{with} \quad \eta_g \nabla \cdot \mathbf{u} + p = 0 \quad \text{in} \quad \Omega_g$$

where $\mathbf{\tau}$ is the viscoelastic extra-stress tensor, $p$ is the pressure, $\eta_g$ is the gas viscosity, $\mathbf{D}$ is the rate of deformation tensor and $\mathbf{I}$ is the unit tensor. Note, that the gas is modeled as a compressible viscous fluid with a bulk viscosity equal to the shear viscosity.

For the polymer, an appropriate constitutive relation is needed which captures the flow behavior in both shear and elongation. In the past decade several models have been proposed which can predict the polymer flow behavior quantitatively. A popular model is the xXtended Pom-Pom (XPP) model [21], adapted by van Meerwald [24]. The XPP model has been derived from the Pom-Pom model [25] which is developed for branched polymers (such as LDPE). The XPP model solves problems of the original Pom-Pom model, such as a zero second normal-stress difference in shear and a discontinuous stress response in elongation.

The three phase contact point where the flow front touches the channel walls is difficult to model [34]. If surface tension can be neglected, which is the case for a melt/air interface [35], the contact angle is 180 degrees, so the flow front approaches the wall tangentially. The flow behavior at the contact point cannot be described by the levelset function [29,30]. It is a reliable, implicit method used in two-component flow simulations of Newtonian liquids [31] as well as non-Newtonian liquids [32,33], for which an additional partial differential equation has to be solved in the entire domain:

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0 \quad \text{in} \quad \Omega$$

where $\phi$ is the levelset function, which represents the signed distance to the interface, so $\phi = 0$ is the interface. The sign of $\phi$ determines the subdomain: $\phi > 0$ is the polymer domain, $\phi < 0$ is the gas domain. Initially $\phi$ is set to the signed distance. However, due to flow $\phi$ does not remain a signed distance function. Therefore a reinitialization step is often applied [30], which regenerates the signed distance function. Since the governing equations of our problem only depend on the sign of $\phi$, the reinitialization step will not be performed in this work.

2.5. Position of the interface

The interface between polymer and gas is computed with the levelset method [29,30]. It is a reliable, implicit method used in two-component flow simulations of Newtonian liquids [31] as well as non-Newtonian liquids [32,33], for which an additional partial differential equation has to be solved in the entire domain:

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0 \quad \text{in} \quad \Omega$$

where $\phi$ is the levelset function, which represents the signed distance to the interface, so $\phi = 0$ is the interface. The sign of $\phi$ determines the subdomain: $\phi > 0$ is the polymer domain, $\phi < 0$ is the gas domain. Initially $\phi$ is set to the signed distance. However, due to flow $\phi$ does not remain a signed distance function. Therefore a reinitialization step is often applied [30], which regenerates the signed distance function. Since the governing equations of our problem only depend on the sign of $\phi$, the reinitialization step will not be performed in this work.

2.6. Contact point modeling

The three phase contact point where the flow front touches the channel walls is difficult to model [34]. If surface tension can be neglected, which is the case for a melt/air interface [35], the contact angle is 180 degrees, so the flow front approaches the wall tangentially. The flow behavior at the contact point cannot be described by the levelset function [29,30]. It is a reliable, implicit method used in two-component flow simulations of Newtonian liquids [31] as well as non-Newtonian liquids [32,33], for which an additional partial differential equation has to be solved in the entire domain:

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0 \quad \text{in} \quad \Omega$$

where $\phi$ is the levelset function, which represents the signed distance to the interface, so $\phi = 0$ is the interface. The sign of $\phi$ determines the subdomain: $\phi > 0$ is the polymer domain, $\phi < 0$ is the gas domain. Initially $\phi$ is set to the signed distance. However, due to flow $\phi$ does not remain a signed distance function. Therefore a reinitialization step is often applied [30], which regenerates the signed distance function. Since the governing equations of our problem only depend on the sign of $\phi$, the reinitialization step will not be performed in this work.

The two dimensionless numbers governing this problem are the viscosity ratio of melt and gas $R = \lambda_b \eta_g / \eta_m$ , the Weissenberg number $Wi = \lambda_b U / 2H$, $U$ is the velocity of the moving wall and $H$ is the height of the channel. All results will be presented in a dimensionless form, where velocities are scaled with $U$, length with $H$ and time with $H/U$.
explained as follows. The wall moves to left, hence the interface in the element attached to the wall will also have moved slightly to the left at the end of a time step. The value of $\phi$ on the wall is reset to zero after every time step, which means that a small amount of gas is continuously being transformed into polymer. As a result, mass conservation of both individual phases are not fulfilled. This is a problem for $n = 0, 1, \ldots$. The time step $\Delta t = t^{n+1} - t^n$ is taken to be a constant, but this is not a requirement. The quantities in the system at the discrete times will also be denoted with a superscript, for example $c^{n+1} = c(t^{n+1})$.

### 3. Numerical methods

The resulting set of equations is solved using a finite element method. Since the problem is non-linear and time dependent, linearization and integration techniques will be used to solve the set of equations. In addition the equations are coupled and solving them simultaneously leads to large systems which require large CPU times to solve [39]. Therefore an operator splitting scheme is used where subsets of equations are solved in a decoupled manner.

The basic procedure is a time-stepping scheme in which the current time $t^n$ and next time $t^{n+1}$ (the one be solved) is defined for $n = 0, 1, \ldots$. The time step $\Delta t = t^{n+1} - t^n$ is taken to be a constant, but this is not a requirement. The quantities in the system at the discrete times will also be denoted with a superscript, for example $c^{n+1} = c(t^{n+1})$.

#### 3.1. Implicit stress formulation

The constitutive equation for the polymer (melt) domain does not have a viscous contribution (solvent). This is a problem for the momentum balance and continuity equation of the polymer domain at the new time $t^{n+1}$ (substitute Eq. (3) into Eq. (1):

$$ -\nabla \cdot \mathbf{u}^{n+1} + \nabla p^{n+1} = 0 \quad \text{in} \quad \Omega_m $$(9)

$$ \nabla \cdot \mathbf{u}^{n+1} = 0 \quad \text{in} \quad \Omega_m $$ (10)

which cannot be decoupled from the constitutive equation to find $(\mathbf{u}^{n+1}, p^{n+1})$. Since a fully coupled procedure would be very expensive, the route given by Bogaerds et al. [40] for the linearized equations and by Davino et al. [41] for the full non-linear equations is followed here. The idea is that first the constitutive equation Eq. (6) in the strong form is discretized in time:

$$ \frac{c^{n+1} - c^n}{\Delta t} + \mathbf{u}^{n+1} \cdot \nabla c^n - (\nabla \mathbf{u}^{n+1})^T \cdot \mathbf{c}^n - \nabla \mathbf{u}^{n+1} \cdot \mathbf{f}_{\text{rel}}(c^n) = 0 $$

(11)

The time-discretization scheme is a first-order semi-implicit Euler scheme where the velocity vector is taken at the new time and the conformation tensor at the current time. From this equation an expression for $c^{n+1}$ can be obtained and, with Eq. (5), an expression for $\mathbf{u}^{n+1}$. The latter is substituted into Eq. (9), the momentum balance for $t^{n+1}$, and the following system is found:

$$ -\nabla \cdot (G \Delta t(-\mathbf{u}^{n+1} \cdot \nabla c^n + (\nabla \mathbf{u}^{n+1})^T \cdot \mathbf{c}^n + \mathbf{c}^n \cdot \nabla \mathbf{u}^{n+1})) + \nabla p^{n+1} $n+1 \quad \text{in} \quad \Omega_m $$

(12)

$$ \nabla \cdot \mathbf{u}^{n+1} = 0 \quad \text{in} \quad \Omega_m $$

(13)

This is a Stokes-like system, linear in $(\mathbf{u}^{n+1}, p^{n+1})$, from which $(\mathbf{u}^{n+1}, p^{n+1})$ can be computed. A weak form of this system can easily be obtained: find $(\mathbf{u}^{n+1}, p^{n+1})$ such that

$$ ((\nabla \mathbf{u})^T, G \Delta t(-\mathbf{u}^{n+1} \cdot \nabla c^n + (\nabla \mathbf{u}^{n+1})^T \cdot \mathbf{c}^n + \mathbf{c}^n \cdot \nabla \mathbf{u}^{n+1})) $$

$$ -((\nabla \cdot \mathbf{u}^{n+1}, \mathbf{p}^{n+1})) $$

$$ = -((\nabla \mathbf{u})^T, G \mathbf{c}^n - \Delta \mathbf{f}_{\text{rel}}(c^n) - I) $$

(14)

for all test functions $\mathbf{v}$ and $q$.
For the gas domain a compressible Stokes system is solved: find \((u^{n+1}, p^{n+1})\) such that

\[
(\nabla \nabla u^{n+1} + \nabla u^{n+1}, \nabla v) - (\nabla \cdot v, p^{n+1}) = 0
\]

\[
(q, \nabla \cdot u^{n+1}) + (q, p^{n+1}) = 0
\]

for all test functions \(v\) and \(q\).

For the trial and test functions of the velocity/pressure, Taylor-Hood \(Q_2 / Q_1\) interpolation is used on quadrilateral elements.

Note, that the systems for the polymer and gas domain are build together and during the integration it is decided which equation is to be resolved by using the log-conformation formulation \([18–20, 44]\).

\[G_{u}^{n+1} = \frac{1}{2}(u^{n+1} \cdot \nabla p^{n+1} + u^{n+1} \cdot \nabla p^{n+1}) = 0 \text{ in } \Omega \]

(22)

For spatial discretization an SUPG stabilization is employed. Finally the weak form can be written as: find \(\phi\) such that

\[\left\{ r + \tau \mathbf{u}^{n+1} \cdot \nabla \phi, \frac{\phi^{n+1} - \phi^n}{\Delta t} + \frac{1}{2}(u^{n+1} \cdot \nabla \phi^{n+1} + u^{n} \cdot \nabla \phi^{n}) = 0 \right\}
\]

(23)

for all test functions \(r\), where \(\tau\) is the same as for the constitutive equation. For trial and test functions of the levelset, \(Q_1\) interpolation has been used on quadrilateral elements.

\[\frac{\partial s}{\partial t} + \mathbf{u} \cdot \nabla s = g(G, s) \quad \text{(19)}\]

Once, \(s\) has been computed, the conformation tensor can be computed by \(\mathbf{c} = \exp s\).

Finally SUPG \([45]\) has been used to stabilize the convection term in the constitutive equation Eq. (19). The weak form reads: find \(s\) such that

\[\left( \frac{d + \tau \mathbf{u} \cdot \nabla d}{\Delta t}, \frac{\partial s}{\partial t} + \mathbf{u} \cdot \nabla s - g(G, s) \right) = 0 \]

(20)

for any test function \(d\). Here, \(\tau = h/2U\), where \(h\) is a typical size of the element in the direction of the velocity and \(U\) is a characteristic velocity magnitude. A semi-implicit first-order time-integration will be used and the final time-discretized weak form becomes: find \(s^{n+1}\) such that

\[
\left( d + \tau u^{n+1} \cdot \nabla d, \frac{s^{n+1} - s^n}{\Delta t} = u^{n+1} \cdot \nabla s^{n+1} - g(G^{n+1}, s^n) \right) = 0
\]

(21)

for any test function \(d\). For the trial and test functions of the conformation tensor, \(Q_1\) continuous interpolation has been used on quadrilateral elements.

The conformation tensor equation is only valid in the polymer domain. In practice the equation for the conformation tensor is solved in the complete domain. However after each time step, the conformation tensor in the gas domain is set to \(\mathbf{c} = \mathbf{I}\) (or \(s = 0\)).
shown, they compare well with results from, for example, Coyle et al. [4].

Next the mesh convergence of the problem is considered. Three meshes (M1, M2, M3) are used, see Fig. 9. In the flow front region the mesh is refined. The amount of elements in this region, the total number of elements and the total number of nodes are given in Table 1.

The shape of the upper half of the flow front is given in Fig. 10 for all three meshes at \( t = 5H/U \). At that time the shape is stationary, and similar for all three meshes. The largest difference can be seen near the contact point, where a finer mesh gives a better approximation of the semi-circular shape. M2 and M3 are converged for the rest of the flow front. In Fig. 11 the time evolution of the velocity in \( x \)-direction is given at \( (x, y) = (3.5H, 0.5H) \). This point is at the interface at \( t = 0 \) and becomes part of the polymer domain for \( t > 0 \). From Fig. 11 it can be observed that the \( x \)-velocity is stationary and equal for the two finest meshes, indicating that the solution

---

**Fig. 7.** The shape of the flow front and particle paths at \( t = 2H/U \) for different viscosity ratios.

**Fig. 8.** Typical deformation pattern of the levelset scalar \( \phi, t = 2H/U \).

**Fig. 9.** The coarsest mesh M1, the initial interface is located at the center of the square central region of the mesh. The domain size \( (x, y) = (6H, H) \).

**Fig. 10.** The shape of the upper half of the flow front for meshes M1, M2 and M3 at \( t = 5H/U \).

**Table 1**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>((N_x, N_y))</th>
<th>Total number of elements</th>
<th>Number of nodal points</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>(26, 26)</td>
<td>2252</td>
<td>9257</td>
</tr>
<tr>
<td>M2</td>
<td>(50, 50)</td>
<td>5276</td>
<td>21413</td>
</tr>
<tr>
<td>M3</td>
<td>(76, 76)</td>
<td>9828</td>
<td>39677</td>
</tr>
</tbody>
</table>
has converged, so mesh M2 will be used from now on. Small oscillations are present in all three solutions. These are explained by the varying leak flow of gas into the polymer domain at the contact point. As the amount of leak flow is determined by the mesh size near the contact point, the amplitude of these oscillations is also expected to be lower for a finer mesh, which is the case, except for $1.5H/U < t < 2H/U$. The reason for the relatively large jumps during this period is unknown.

4.2. Viscoelastic flow

Viscoelastic flow simulations have been carried out using different Weissenberg numbers. In order to perform these simulations the non-linear parameters have to be chosen. Some combinations of relaxation times and non-linear viscoelastic parameters result in unphysical behavior in shear and/or elongation, such as too much shear thinning and maxima in the steady state shear stress. From Fig. 12 the steady state shear response of the XPP model for different parameter sets is shown. It can be seen that the steady state shear stress shows a maximum and minimum in the shear stress for $r = (4, 6)$ and $q = (5, 9, 13)$. This is considered as unphysical behavior, since polymer melts in general show a shear stress that is continuously increasing or at least constant for increasing shear rates. Shear flow with $(r, q) = (2, 5)$ does not show this behavior and therefore this parameter set will be used.

With the material properties fixed, first the shape of the flow front is investigated for different Wi, since from [17] it is known that for a Weissenberg number that is lower or higher than unity the flow front shape will be flattened. In Fig. 13 the flow front shape is shown for Wi = 0.1, 1.0, 2.5, 5.0, 10.0 at $t = 50H/U$. In agreement with Bogaers et al. [17], both for lower and higher Wi than unity the flow front is flattened. For Wi = 10, the flow becomes unstable, hence an asymmetric flow front is observed.

4.3. Unstable fountain flow

The unstable motion of the flow front is further investigated by computing the behavior of $u_y$ at the center line of the channel. For stable flow, this velocity component is zero everywhere on the center line. Any deviation from zero, is an indication that the flow is asymmetric. In order to initialize the possible slow growth of the asymmetric flow, a random disturbance of order $10^{-3}$ is added to the initial fully relaxed conformation tensor, similar to [47]. The velocity at $(x, y) = (3.5H, 0.5H)$ is shown. The results are given for Wi= 0.1, 1.0, 2.5, 5.0, 10.0, see Fig. 14.

The flow of XPP is stable for Wi= 0.1, 1.0, 2.5, since $u_y$ only shows numerical noise due to the varying leak flow. For Wi= 5.0, the flow is stable too, but the appearance of two entrapped “fictitious gas” bubbles causes a asymmetrical motion at $t = 33H/U$, which disappears again afterwards. For Wi= 10.0, however, the flow is unstable, resulting in an oscillating velocity in $y$-direction. Notice that the mean velocity in $y$-direction is negative, indicating that the center line fluctuation is not symmetrical around the initial position. At $t = 26H/U$, two spurious entrapped bubbles of the “fictitious gas” appear in the melt, causing large jumps in the velocity in $y$-direction. After the “bubbles” are convected away from the flow front region, the oscillatory behavior is regained. Since the oscillating motion is present before the occurrence of the gas “bubbles”, the spurious gas is not the cause of the unstable motion. The growth of the instability is shown in more detail in Fig. 15, where the norm of the velocity in both the $x$- and $y$-direction are given. Due to the random disturbance to the initial conformation tensor, a disturbance of $10^{-3}U$ of the $y$-velocity at the centerline is present after the first time step. First a linear growth of $u_y$ is observed until $t = 7H/U$, after which the oscillating behavior shows up. It should be noted here, that based on the center line velocity, the time needed for particles to flow from the piston to the flow front is at least $10H/U$. This is beyond the time the actual flow front dynamics sets in (see Fig. 15). Therefore it can be assumed that the piston is positioned far enough to have negligible influence on the local flow front dynamics.

The model shows high frequency oscillations in $u_y$ for both stable and unstable flow conditions. The amplitude of these oscillations decreases with increasing Wi. This is explained by the relatively large momentum gain from the artificial creation of melt due to leak at the interface for low Wi. For higher Wi this moment-

![Fig. 11. Velocity in x-direction for meshes M1, M2 and M3 at $(x, y) = (3.5H, 0.5H)$.](image1)

![Fig. 12. Steady state shear viscosities for single mode XPP, for $r = 2$ (top), $r = 4$ (middle, shifted down one decade), and $r = 6$ (bottom, shifted down two decades). For each relaxation time ratio four different $q$ are given.](image2)

![Fig. 13. The flow front shape for Wi= 0.1, 1.0, 2.5, 5.0, 10.0, for XPP. The grey unsymmetrical shape in the right figure is for Wi= 10.](image3)
tum gain is relatively less important, since the polymer stresses are much higher.

The main frequency of the unstable motion is about $0.3U/H$, which can be found from the power spectrum of $u_y$ in $(x, y) = (3.5H, 0.5H)$, see Fig. 16. The power spectrum gives the energy distribution for all frequencies present in $u_y$. Most energy is present at low frequencies, with a maximum at $0.33U/H$ and at higher frequencies which are multiples of the lower frequencies. This results in a typical length scale of the repeating motion of $3H$, which is in good agreement with experiments by Bulters et al. [16], where the repeating distance is three to ten times the height of the channel.

In addition to the frequency, the direction of the motion disturbance is important. In order to visualize the relatively small disturbance, the difference in velocity between the upper and lower half is shown in Fig. 17, at $t = 21.4H/U$.

From Fig. 17, the swirling motion can be observed in a large part of the melt region, also observed by Bogaerds et al. [17]. Near the front however, the flow is reversed. This can partly be explained by the fact that the flow front is not symmetrical at this time. Therefore opposing points near the interface, can be of different material, which results in a comparison between the velocity in the gas with the velocity in the melt. Bogaerds et al. [17] performed a linear stability analysis, thus predicting the critical Wi and the most unstable eigenmode of the flow. In this work all non-linear terms are taken into account and in addition to the linear growth of the instability, the non-linear oscillating motion is observed. In [17] the critical Wi for $(r, q) = (2, 5)$ is 2.5, the growth rate is very small however. Although simulations of $Wi > 2.5$ have not been performed by Bogaerds for this particular parameter set, it is expected that the flow at higher Wi will be unstable too. In our study only $Wi = 10$ shows the unstable motion. A reason that the critical Wi for the non-linear computations is higher than for the linear instability simulations might be the different modeling of the interface, including the contact point, using a level set function compared to the height function used in [17]. Further investigation of the comparison between the linear and non-linear instability will be part of future work.
Fig. 15. The evolution of both $u_x$ (top) and $u_y$ (bottom) at $(x, y) = (3.5H, 0.5H)$, $Wi=10$.

Fig. 16. Power spectrum of $u_y$ at $(x, y) = (3.5H, 0.5H)$ for $10H/U < t < 50H/U$.

Fig. 17. The difference between velocity in the upper and lower half of the channel at $t = 21.4H/U$.

Fig. 18. Particle paths of 20 particles at $t = 21.4H/U$. The flow front is the right curve.

In order to illustrate the asymmetrical fountain flow, the paths of 20 particles are shown in Fig. 18. The asymmetrical motion of the fountain flow is clearly visible.

Due to the large deformations during one simulation (material makes approximately 3–4 cycles from piston to flow-front), the gradients in the levelset function become large and cannot be captured well anymore. This leads to interface generation, especially near the contact point and near the piston corners. A likely solution to this problem is the reinitialization of the levelset function to a signed distance function at regular intervals.

5. Conclusions

The viscoelastic flow front instability has been studied by numerical simulation. A two-phase viscoelastic model in two dimensions has been developed which predicts a fountain flow instability and is able to monitor this instability in the full non-linear regime. The modeling of the three phase contact point has been addressed and solved by modeling the gas as a compressible viscous fluid and treating both flow front and wall as an interface. An operator splitting scheme is applied in such a way that a stable set of equations is obtained without the need for additional viscous solvents. The final set of equations is solved with a finite element method. The discontinuous properties over the interface are captured well, by applying local spatial refinement of the integration points. At $Wi = 10$ a periodic motion of the flow front is observed with a spatial period of three times the channel height, which is in agreement with experiments. This is an extension of the model developed by Bogaerds et al. [17], where only the onset of this instability was studied.

References


